FLOW DETECTION ON MANIPULATOR ARM ROV BY USING CFD

N. Asmuin*, K. Ngadimon, W.L. Mak
Dept. of Plant and Automotive Engineering
Faculty of Mechanical Engineering & Manufacturing
Universiti Tun Hussein Onn Malaysia (UTHM)
norzela@uthm.edu.my, knizam@uthm.edu.my, wlmark88@gmail.com

Keywords - Remotely Operated Underwater Vehicle, (ROV), Computer fluid dynamis,CFD.

Abstract-A remotely operated vehicle (ROV) is a tethered underwater robot in offshore industry. As the oil and gas industry moves to more subsea environment, ROVs will become an even more important in drilling, development and repair offshore. The pahl and Beitz method is using to design a functional manipulator arm. Some of the sensitivity equipment like sensor needs to be installed on the manipulator arm to improve the performance. Analyze the manipulator arm under various fluid flow conditions was done by using computational fluid dynamics CFD ANSYS software. Simulation is done on the manipulator arm model drawn in the Solidworks Design Application software. Simulation on the manipulator arm is carried out with different yaw angle of 0°, 15° and 30° and different depth in 1m, 5m and 10m. The inlet is set to 1.5m/s and outlet is set to 1.3m/s. Pressure regions acting on the manipulator arm have the lower velocity. This means that pressure is inversely proportional to the velocity. Reynolds number, pressure and velocity contours determined by using CFD software.

Introduction

Grabber and manipulator arm is one of the main parts on remotely operated underwater vehicle (ROV). The manipulator arm is attached to the front –middle –bottom on the ROV. Manipulator arm are usually programmable, with similar function to a human arm where the manipulator arm are connected by joint and allow move in rotational or translational motion. The different in this design is the manipulator arm on ROV functions underwater. Thus, waterproofing is needed. The manipulator arm is programmable and connected to the controller where operators on the surface can navigate it through umbilical cable and controller. The objective of this study is to design a suitable manipulator arm with gripper using solid work and analyze the manipulator arm with gripper under various fluid flow conditions such as fluid flow analysis, pressure and velocity contour using CFD ANSYS. The fluid condition is incompressible fluid. According to Pahl and Beitz method [1], the identification of need is based on the four criteria such as ease of water-resistance, function of manipulator arm, cost and weight.

Low-cost underwater manipulator arm has been developed by Lauren [2]. The manipulator arm is consisting of five degree of freedom for a small ROV. The primary achievement is the design of a waterproof servo motor housing consisting of static and dynamic seals and proven to be watertight, and to have minimal effect on motor performance. The material of housing to make the motor become water proof, the servo is using black nylon 6/6 where it is low cost, good tensile strength and dimensional stability. The material of linkages where to connect with joint (housing) is made of clear polycarbonate. While theory and design issues of underwater manipulator has been discussed by Irfan [3]. He found that the hydrodynamics equation has been successfully integrated with the normal manipulator dynamics from his experiment. The effect of each of this parameter towards the torque requirement of the joint can be determined. Therefore, this will help us to understand what the torque equipment will be to generate the design criteria of the actual manipulator.

Pressure contours, velocity vector and velocity streamlines for any vehicles can be determine by using the CFD simulation. Therefore, a full aerodynamic survey on national car was simulated by using the CFD software [4]. The simulation tested in different yaw angle from 0° to 30°. CFD also was used on analysis airflow around the airship [5]. The airship model set at wind velocity of 24km/h and simulating in different yaw angles and angle of attack [6]. The lift, drag and moment coefficient can be determined by using CFD software. CFD software is expected to provide Reynolds number with very accurate information to the manipulator arm ROV.

Methodology

There are 3 conceptual designs has been made based on the characteristic of each method observed. The first conceptual of manipulator arm is by using PVC pipe. The manipulator arm is two link arms with two degree of freedom in X-direction. The link and joint are connected by DC servo motor where using gear and shaft system to transmit torque. The grabber is also using gear and shaft system. The second conceptual of manipulator arm is using aluminium material. The manipulator arm is one link arm where can move in two degree of freedom in X and Y-direction. Power supply is battery. The link and joint are connected with DC servo motor where using gear and shaft system to transmit torque. The grabber is using the electromagnetic theory to control the grabber open and close. The third conceptual manipulator arm is made of polycarbonate. The manipulator design in two link arm where can move in three degree of freedom. Power supply is battery. The link and joint are connected with DC servo motor where transmit torque. The grabber is using gear and shaft system to control the grabber to open and to close hand.

Simulation process was using Computational Fluid Dynamic (CFD) ANSYS software. CFD is a tool to solve fluid flow problem. CFD in practice can be divided into four categories which are: problem identification, pre-processing, solver and post-processing. Pre-processing consist of 4 main parts: Geometry creation, Meshing Domain, physics and solver settings. The manipulator arm model was design in SOLIDWORK software. In this process, the geometry for transient structural is linked with fluid flow (ANSYS CFX). After modelling process, the model has to be meshed by using CFX-Mesh in ANSYS. The manipulator arm model is meshed using body sizing where the element size is 2mm. The fluid region is using the patch conforming method which is tetrahedrons. The velocity for the inlet of the flow domain was set as 1.5m/s, and the velocity at the outlet of the flow domain is 1.3m/s. The wall is set to no slip boundary condition. The turbulent model set to the RNG k-Epsilon. The referent pressure is set into three conditions where to represent the different of depth such as 1m, 5m, and 10m. The different of depth will tested in different of yaw angle such as 0°, 15° and 30°. The result in post-processing has been shown by viewing the contour, vector and streamline of effects like velocity, pressure and temperature on a manipulator arm model.

Result and discussion

The pressure in different depth is calculated shown in Table 1. The maximum pressure and minimum pressure in different depths and different angles are shown in Table 2, Table 3 and Table 4. From the three tables below, we can conclude that the value of maximum pressure acting on the manipulator arm is almost the same (1170.37Pa to 1193.48Pa) in different depths and different angles. The minimum pressure acting on the manipulator arm is in the range -4895.77Pa to -5621.47Pa for three different depths and different angles.

Table 1: Various pressure with various height

Depth , h (m)	Gravity , g (m/s^2)	Density of fluid (kg/m^3)	Pressure, P (Pa)
1	9.81	1000	9810
5	9.81	1000	49050
10	9.81	1000	98100

Table 2: Pressure in different depths at yaw angle 0° with inlet velocity 1.5 m/s and outlet velocity 1.3 m/s

Angle (°)	Depth (m)	Inlet (m/s)	Outlet (m/s)	Max Pressure	Min Pressure
0	1	1.5	1.3	1184.12Pa	-5339.27Pa
0	5	1.5	1.3	1184.06Pa	-5297.51Pa
0	10	1.5	1.3	1184.02Pa	-5293.86Pa

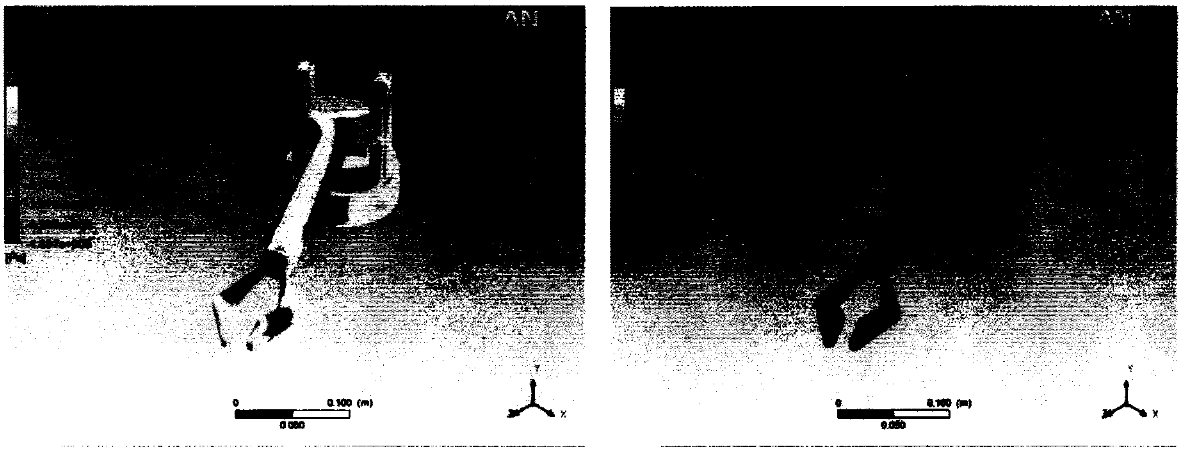
Table 3: Pressure in different depths at yaw angle 15° with inlet velocity 1.5 m/s and outlet velocity 1.3 m/s

Angle (°)	Depth (m)	Inlet (m/s)	Outlet (m/s)	Max Pressure	Min Pressure
15	1	1.5	1.3	1170.37Pa	-5538.84Pa
15	5	1.5	1.3	1170.43Pa	-5572.02Pa
15	10	1.5	1.3	1170.42Pa	-5621.47Pa

Table 4: Pressure in different depths at yaw angle 30° with inlet velocity 1.5 m/s and outlet velocity 1.3 m/s

Angle (°)	Depth (m)	Inlet (m/s)	Outlet (m/s)	Max Pressure	Min Pressure
30	1	1.5	1.3	1187.33Pa	-4997.00Pa
30	5	1.5	1.3	1187.15Pa	-4895.77Pa
30	10	1.5	1.3	1193.48Pa	-4897.09Pa

The red region in the Fig. 1 (a) has shown the high pressure acting at the front of the Manipulator arm due to the collision with the fluid. The yellow region start indicates a negative pressure distribution. The negative pressure indicated that the force is acting along the opposite surface normal. This is the part to undergo the high velocity. Due to the Boyle's law, the high pressure region will undergo the low velocity. The high and low pressure is acting on the same region of the manipulator arm model in different depths. Fig. 1(b) shows the high velocity (red region) acting on the two side edge of based manipulator arm model where it is front edge of the supporter bar housing to the shaft of manipulator arm. The maximum value acting on the model in 10m depth is 3.028m/s. Minimum velocity (blue region) acting on the front of the manipulator arm. The minimum value acting on the model in 10m depth is 0.01122m/s.



(a) (b)

Fig. 1: (a) Pressure contour and (b) velocity contour

Table 5: Velocity in different depths at yaw angle 0° with inlet velocity 1.5 m/s and outlet velocity 1.3 m/s.

Angle (°)	Depth (m)	Inlet (m/s)	Outlet (m/s)	Max Velocity	Min Velocity
0	1	1.5	1.3	3.05178m/s	0.0059211m/s
0	5	1.5	1.3	3.0359m/s	0.00502671m/s
0	10	1.5	1.3	3.0275m/s	0.0111216m/s

Table 6: Velocity in different depths at yaw angle 15° with inlet velocity 1.5 m/s and outlet velocity 1.3 m/s.

Angle (°)	Depth (m)	Inlet (m/s)	Outlet (m/s)	Max Velocity	Min Velocity
15	1	1.5	1.3	3.12289m/s	0.01144861m/s
15	5	1.5	1.3	3.1334m/s	0.0166004m/s
15	10	1.5	1.3	3.14831m/s	0.0145322m/s

Table 7: Velocity in different depths at yaw angle 30° with inlet velocity 1.5 m/s and outlet velocity 1.3 m/s.

Angle (°)	Depth (m)	Inlet (m/s)	Outlet (m/s)	Max Velocity	Min Velocity
30	1	1.5	1.3	3.02672m/s	0.00652266m/s
30	5	1.5	1.3	3.01019m/s	0.0152808m/s
30	10	1.5	1.3	3.01319m/s	0.0167631m/s

The maximum velocity and minimum velocity in different depths and different angles are shown in Table 5, Table 6 and Table 7. From the three tables below, we can conclude that the value of maximum velocity acting on the manipulator arm is almost the same (3.01019m/s to 3.1483m/s) in different depths and different angles. The minimum value velocity acting on the manipulator arm is in the range 0.0050267m/s to 0.0167631m/s. The maximum and minimum value for the velocity is acceptable. Velocity vector can be determined and the motion of fluid flow be detected. It will show the direction of motion in line arrow pattern. The results of Reynolds number are shown in Table 8, Table 9 and Table 10.

Table 8: Reynolds number with yaw angle 0° at point 1, 2, 3 and 4

Point	Reynolds number at 1m depth ($X10^5$)	Reynolds number at 5m depth ($X10^5$)	Reynolds number at 10m depth ($X10^5$)
1	4.374	4.314	4.334
2	3.070	2.982	3.402
3	4.186	3.069	1.706
4	4.860	4.800	4.577

Table 9: Reynolds number with yaw angle 15° at point 1, 2, 3 and 4

Point	Reynolds number at 1m depth ($X10^5$)	Reynolds number at 5m depth ($X10^5$)	Reynolds number at 10m depth ($X10^5$)
1	4.489	4.505	4.443
2	0.8888	0.9177	1.485
3	3.240	3.303	2.665
4	4.556	4.503	4.866

Table 10: Reynolds number with yaw angle 30° at point 1, 2, 3 and 4

Point	Reynolds number at 1m depth ($X10^5$)	Reynolds number at 5m depth ($X10^5$)	Reynolds number at 10m depth ($X10^5$)
1	1.216	1.098	0.6449
2	1.683	2.372	1.753
3	1.291	1.459	2.903
4	1.715	1.326	1.158

(a) (b)

Fig. 2: (a) Reynolds number detect at Point 1 and Point 2 and Reynolds number detect at Point 3 and Point 4

The Reynolds number at point 1 and point 2 shown in Fig. 2(a), from the result shown that all the point is still undergo laminar flow. The Reynolds number at point 1and point 2 shown in Fig. 2(b). The turbulent flow happens when the Reynolds number is more than 500000. The Reynolds number formula was input in CFD software to determine the value of Reynolds number at point 1, 2, 3 and 4.

Conclusion

CFD analysis has been successfully conducted to investigate flow around the manipulator arm model. The pressure and velocity contour are obtained from CFX-solver as shown in Fig. 2(a) and Fig. 2(b). The value of pressure and velocity contour is acceptable. CFD can used to predict the external fluid flow of manipulator arm. Vortex happens behind the manipulator arm model. The yaw angle of manipulator arm model change will influence the streamline flow. The different angle of flow must be tested to get the average data for the overall performance of manipulator arm model. Based on the Reynolds number, the suitable place to put the sensor is put on the left or right hand side of the supporter bar housing to the shaft of manipulator arm where it has the low velocity. Therefore, no sensor and electrical box can put at vortex region. The high velocity of fluid flow will influence the sensitivity of the sensor. It needs some improvement on the design of manipulator arm and the simulation process. Some of the recommendations is using the fluid solid interface or moving mesh method to run the simulation. This method can improve accuracy of the data where the manipulator arm and the fluid are moving simultaneous. It can determine every motions of the manipulator arm under fluid flow condition.

ACKNOWLEDGEMENT

We wish to thank the Centre for Energy and Industrial Environment Studies (CEIES), Universiti Tun Hussein Onn Malaysia (UTHM), Parit Raja, Batu Pahat, Johor, Malaysia for their support.

REFERENCES

[1] G.Pahl, W. Beitz, J. Feldhusen, K.H. Grote. (2007). Engineering design, Third Edition. Warren Seering, Weber-Shaughness Professor of Mechanical Engineering, Massachusetts Institute of TechnologySpringer. ISBN 978-1-84628-318-5.

[2] Cooney L.A..(2006). Development of a Low-Cost Underwater Manipulator, Massachusetts institute of technology : *Bachelor of Science.*

[3] Abdul Rahman, I., Mat Suboh, S., Arshad M. R. (2007). Theory and design Issues of Underwater Manipulator, Instrumentation and Mechatronics Engineering. *University Sains Malaysia. International Conference on control.*

[4] MOHD AMIN, N.A. (2005). Aerodynamic Survey on a Passenger Car using CFD, Faculty of Mechanical Engineering and Manufacturing. University Tun Hussein Onn Malaysia.

[5] KHALIL, M.A. (2006). Study and Aerodynamics Testing of an Airship using CFD, Faculty of Mechanical Engineering and Manufacturing. University Tun Hussein Onn Malaysia.

[6] Arshad, M.R. (2011). Short –course on Underwater Vehicle Technology. Underwater Robotics Research Group (URRG). (3^{rd} until 5^{th} October 2011).